# *Chapter: 11*

# MEK4560
# The Finite Element Method in Solid Mechanics II

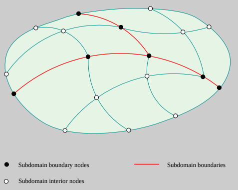
(April 9, 2008)


Torgeir Rusten

(E-post:torgeiru@math.uio.no)



Department of Mathematics
University of Oslo





Contents

Page 1 of 38

Go Back

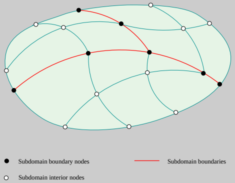
Close

Quit

# Contents




Department of Mathematics
University of Oslo



Mek 4560
Torgeir Rusten

# 11. Nonlinear analysis, part I

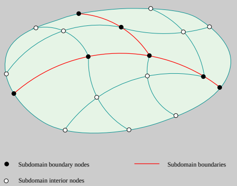
This is the first of two Chapter discussing nonlinear analysis. They introduce the two main areas in nonlinear analysis, *formulation* and *solution of the nonlinear algebraic equation* resulting from a finite element method:

1. *Formulation*: Discuss the continuum mechanical equations used in nonlinear structural analysis. It is include in order to give some insight into the foundation of nonlinear Finite Element software.

2. *Solution*: The nonlinear algebraic equations arising from a using a Finite Element method to discretize the nonlinear models. In Chapter 12 a discussion of some numerical methods is included.

Linear Finite element analysis are well understood and can be used as "black box" technology, i.e. it can be used effectively with knowledge of the mathematical models of linearized elasticity together with basic insight in the Finite Element method. Detailed knowledge of the linear solvers and eigenvalue methods are not required in most cases. Nonlinear analysis is not as well understood, and a deeper insight into the physical effects, mathematical models and numerical methods are necessary in order to do reliable analysis of constructions where nonlinear effects are important. (This might be a motivation to study linear elasticity in some detail!)

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

Contents

Page 3 of 38

Go Back

Close

Quit

The following example is illustrative:

**Linear analysis:** A linear system (satisfying a few simple assumptions) has a unique solution.

**Nonlinear analysis:** Nonlinear algebraic equations can have a large number of solutions:

> 100 cubic equations has $3^{100} \approx 10^{47}$ solutions. (The number of atoms in the known universe is estimated to be $10^{50}$!).

Ideally one would like to derive models that has only one solution, but sometimes there are several solutions that are physically meaningful. Sometimes the mathematical model has unphysical solutions, ideally the model should be reformulated in order to reflect the physics better but sometimes the model at hand is the best available. How can we find the relevant solutions, how to avoid spending computational time on unphysical solutions?

We discuss this briefly in Chapter 12.

In Chapter 10 linearized buckling for a beam was introduced. In the present chapter we generalize the expressions derived in the beam example. We'll see below that the buckling equations are a linearization of a set of nonlinear equilibrium equations.

Department of Mathematics
University of Oslo

In [Cook et al., 2002][1], Chapter 17 discuss nonlinear analysis.

**Nonlinear equilibrium equations:** We outline a derivation of the equilibrium equations for nonlinear analysis. Application of the Finite element method result in a set on nonlinear algebraic equations, the solution to these equations are (hopefully) an approximation to the continuous solution.

## 11.1. Nonlinear effects, a visual tour

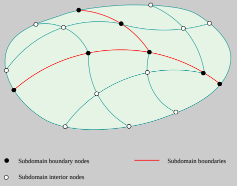
The figure below show the four types of nonlinear effects

$$\textit{nonlinear}\begin{cases} \textit{geometry} \\ \textit{material} \\ \textit{boundary conditions} & \begin{cases} \textit{kinematic} \\ \textit{loads} \end{cases} \end{cases}$$

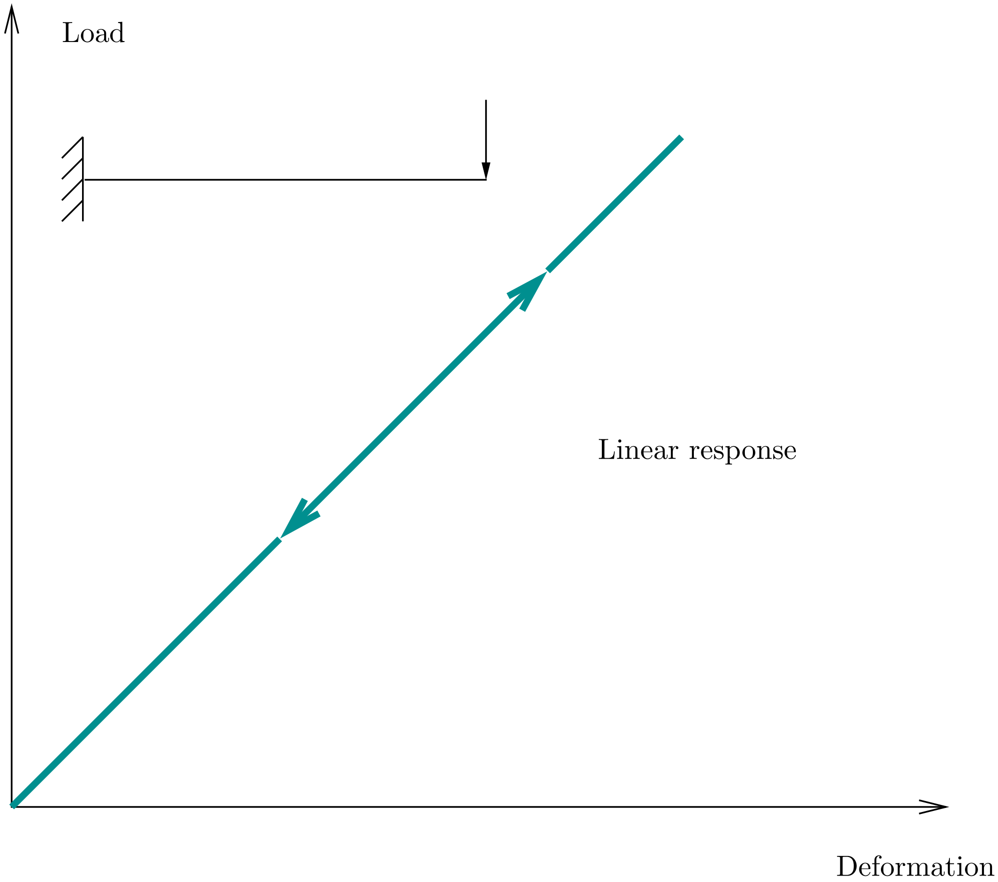
in structural analysis. The figures below show some typical nonlinear behavior.

[1] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt. *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition, October 2002.

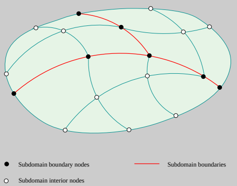
Department of Mathematics
University of Oslo

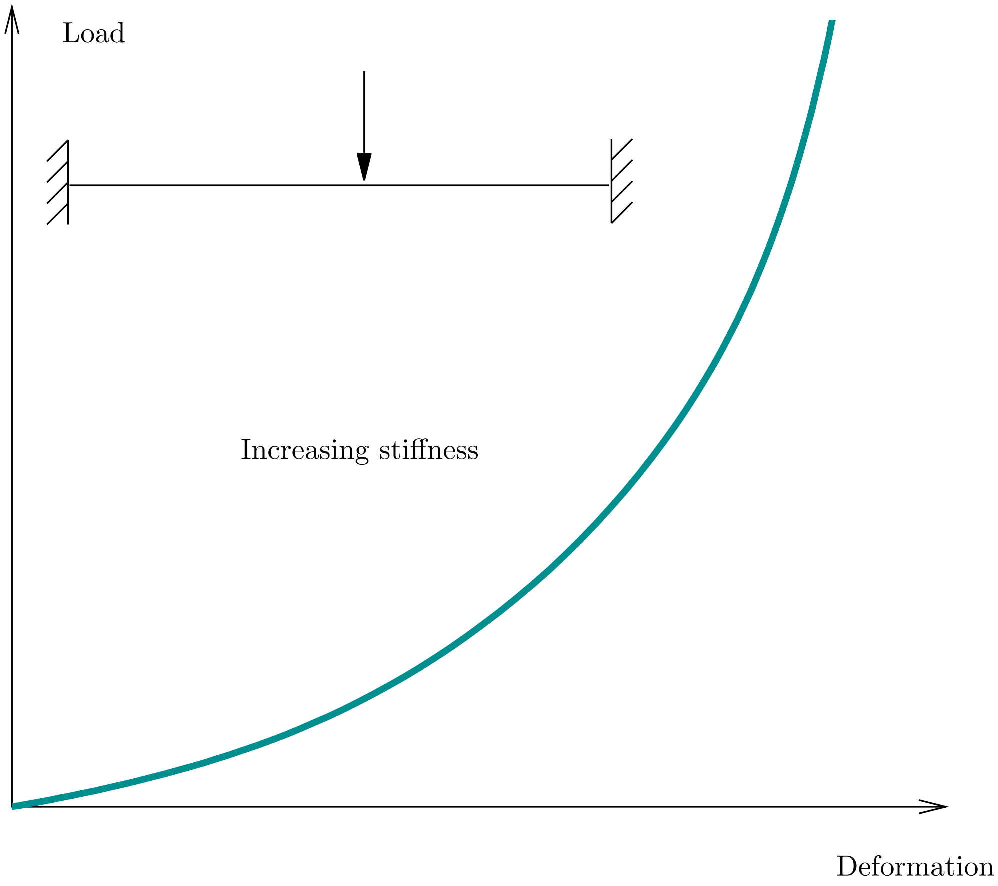



Mek 4560
Torgeir Rusten


Load
Linear response
Deformation

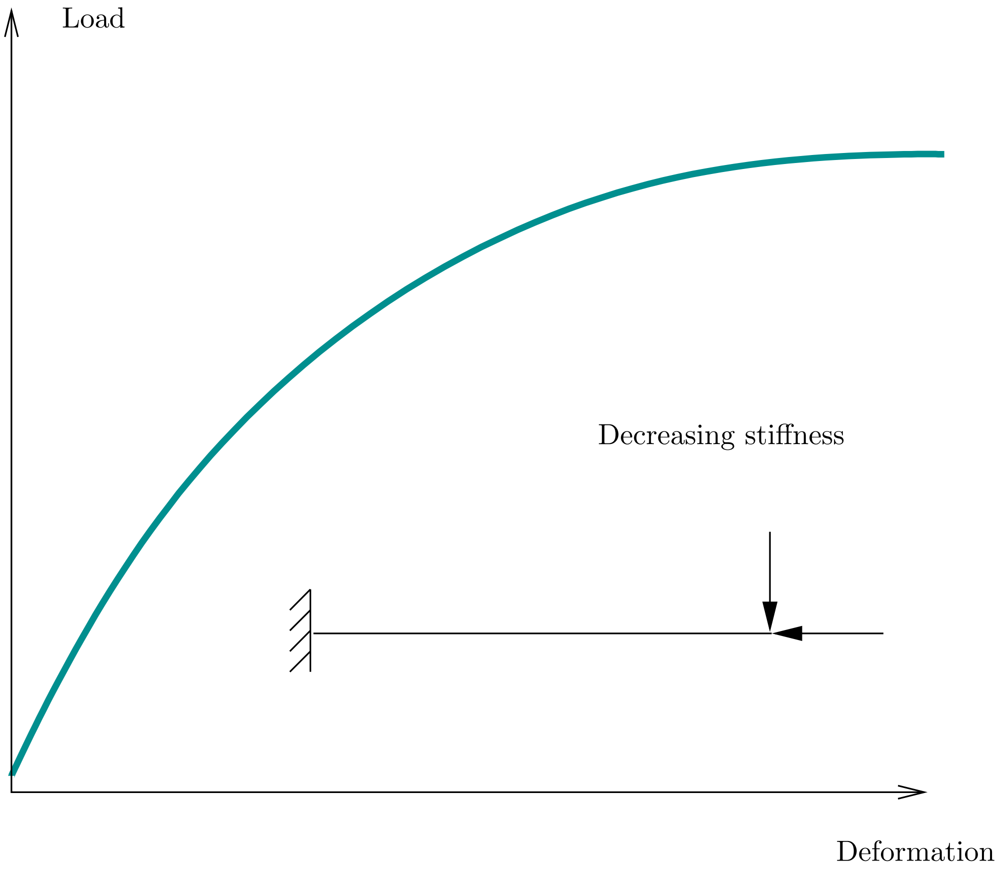



Subdomain boundary nodes
Subdomain boundaries
Subdomain interior nodes




Department of Mathematics University of Oslo

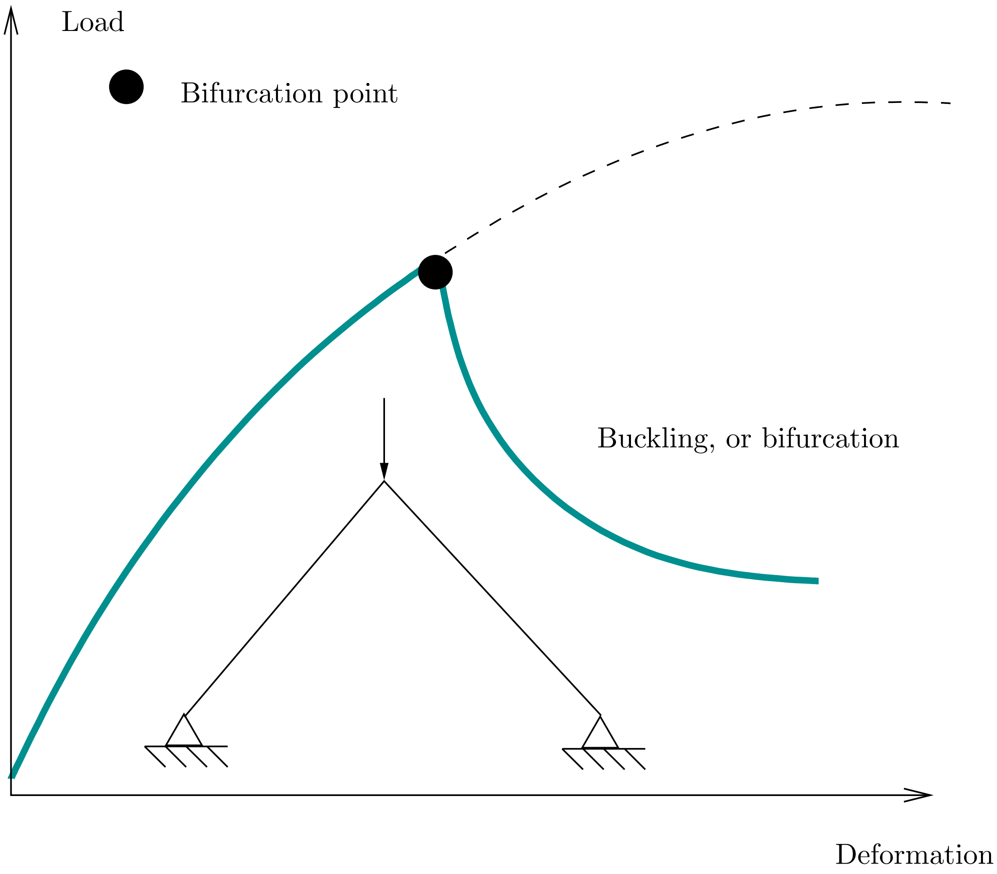
Mek 4560
Torgeir Rusten




Department of
Mathematics
University of Oslo

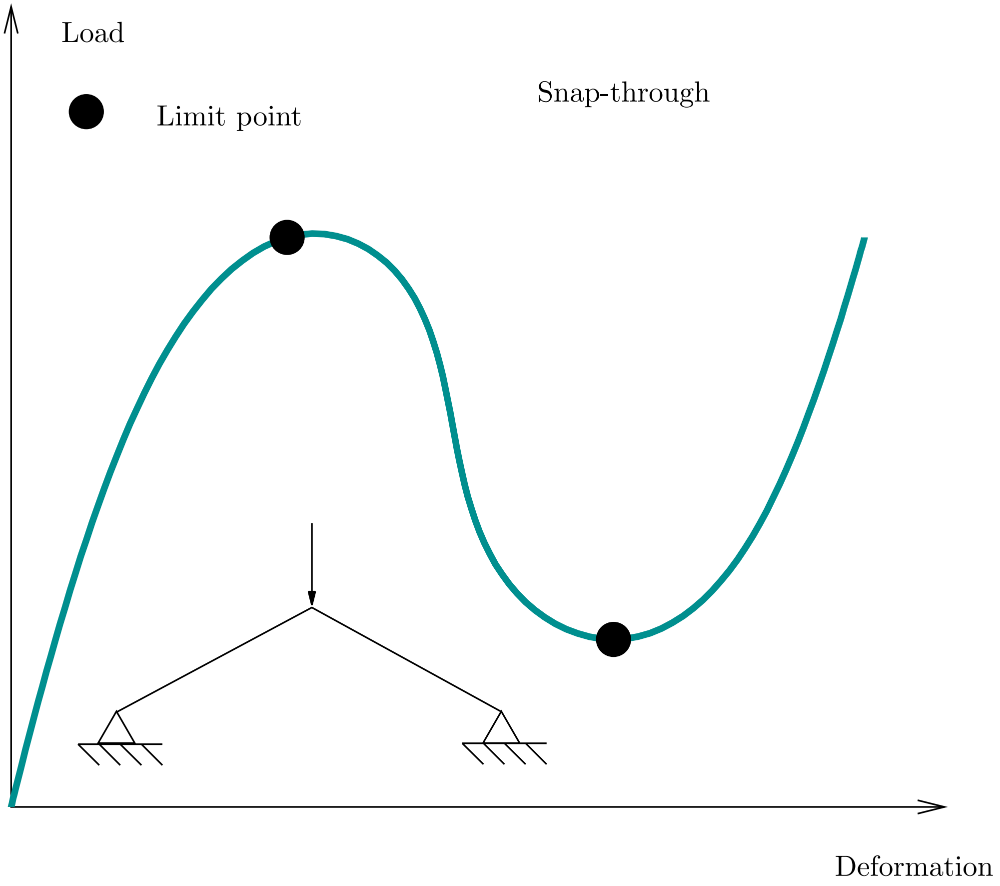


Mek 4560
Torgeir Rusten

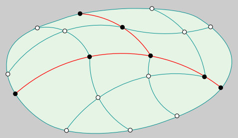



Department of Mathematics
University of Oslo

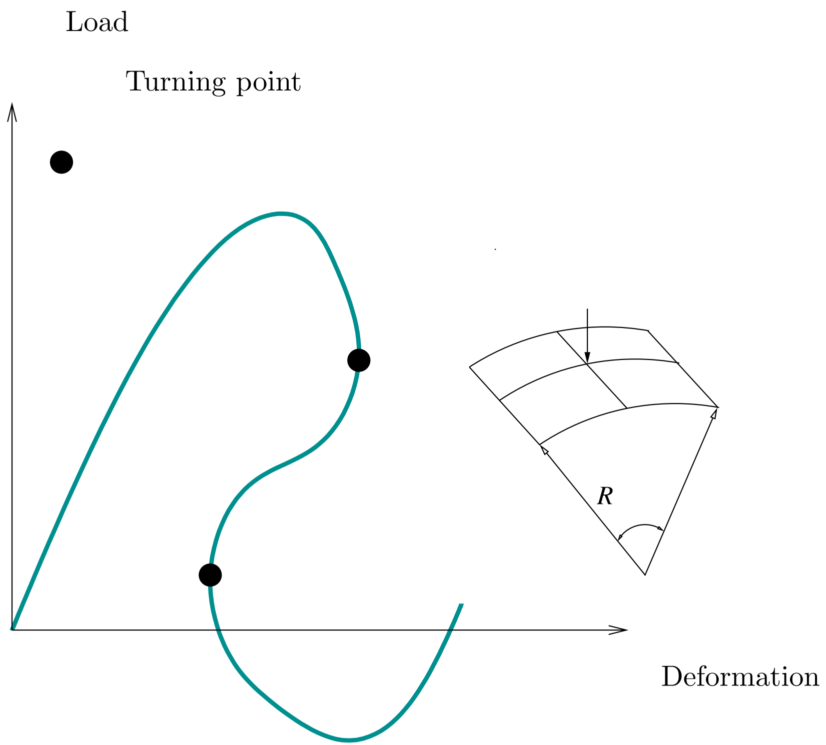
Mek 4560
Torgeir Rusten

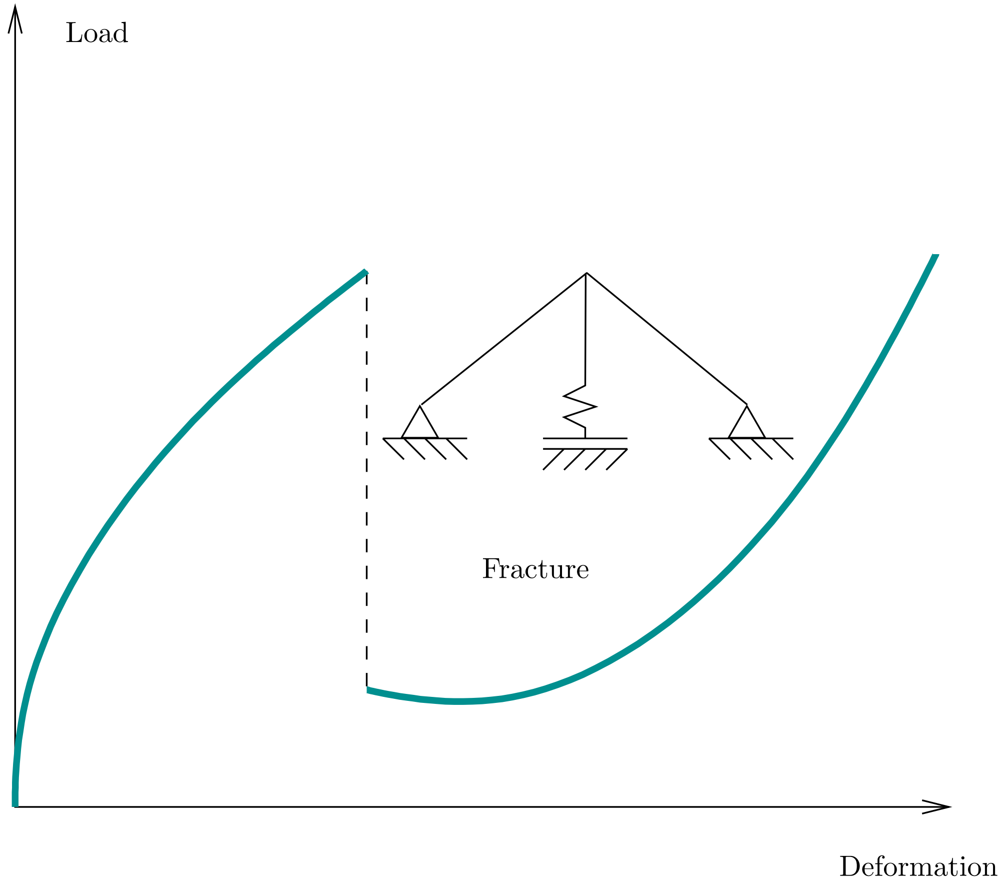



Department of Mathematics
University of Oslo

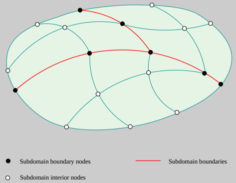



Mek 4560
Torgeir Rusten


Load
Turning point
Deformation
R




Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

## 11.2. Continuum mechanics

We consider a continuum and its deformation in a *Cartesian coordinate system*[1]. In linear analysis the Finite element methods was based on the principle of *virtual work* or, equivalently, the the principle of *minimum potential energy*. Nonlinear models discussed here are based on the same principles, but in deformed geometry:

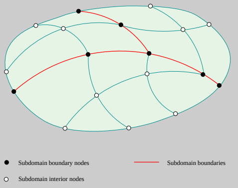
$$\delta\Pi(\boldsymbol{u};\delta\boldsymbol{u}) = \int_{V(t)} \boldsymbol{\sigma} : \delta\boldsymbol{\varepsilon}\, dV - \int_{V(t)} \boldsymbol{F}\cdot\delta\boldsymbol{u}\, dV - \int_{S(t)} \boldsymbol{\Phi}\cdot\delta\boldsymbol{u}\, dS = 0 \tag{11.1}$$

The parameter $t$ is a (pseudo) time variable. In order to analyze large deformations it is necessary to consider the deformed geometry because the the nonlinearity is due to the change in geometry due to the loads. In the Finite element method the equations are frequently transformed back to the undeformed geometry. A linearization of the nonlinear equilibrium equations result in equations similar to the equations used in linearized buckling. The linearization is also used in some numerical methods for the solution of systems of nonlinear algebraic equations, in particular Newtons method.

In order to introduce the equilibrium equations, we review of kinematics; the description of a body's deformations.

[1] I.e. we do not use curvelinear coordinates

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

Contents

Go Back
Close
Quit

### 11.2.1. Kinematics

The body, or construction, is assumed to be a continuum, denoted $\mathcal{B}$ and consists of a set of particles, $\mathcal{P}$, with mechanical properties. A particle is identified by a set of Cartesian coordinates, $(x, y, z) = \boldsymbol{x}$. The particles move when external forces are applied to the body and the movement is described by the displacements $\boldsymbol{u}(\mathcal{P}) = \boldsymbol{u}(\boldsymbol{x}, t)$. The displacement of all the particles is a displacement field:

$$\boldsymbol{u} = \boldsymbol{u}(\boldsymbol{x}) \qquad \boldsymbol{x} \in \mathcal{B}$$

The displacement field must be ....(legal):

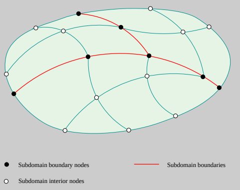
1. Continuous $\rightarrow$ no "holes".
2. Kinematical constraints must be satisfied.

The kinematical description is important in order to be able to follow the construction when loads are applied, we use a *kinematic description* relative to the *reference configuration*.

In order to implement a nonlinear Finite Element method we must make up our mind regarding:

- A *Lagrangian* or *Eulerian* description of kinematics?
- A reference configuration.

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

**Kinematics:** The are two methods for modeling kinematics:

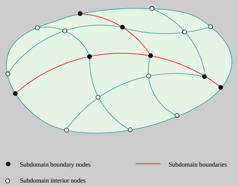
1. *Lagrangian*: We follow the particles in the continuum. This is common in solid mechanics. A Finite Element mesh follow the movement.
2. *Eulerian*: We observe quantities at a fixed point in space. This is common in fluid mechanics. A Finite Element mesh is fixed.

**Reference configuration:** Here we discuss the Lagrangian description. Four different reference configurations are used in Finite Element programs for structural analysis:

1. *Total Lagrangian formulation:* The reference configuration is the initial configuration and stress and strains are relative to this position.
2. *Updated Lagrangian formulation:* The reference position is the current equilibrium position and stress and strains are relative to this position.
3. *Corotational formulation:* Two reference configurations are used. Stress and strains are relative to the new equilibrium position while rigid body movements are relative to the initial configuration. For analysis of structures with small strains and large rotations, a linear element method can be used in the deformed configuration.
4. *Deformed formulation:* All quantities relative to the deformed configuration.

All the description are equivalent, the choice is based on convenience. However, it is important to be aware of the formulation used in the analysis software!

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

The alternative formulation result in different formulations of stress and strains.

Henceforth we consider the *Total Lagrangian formulation.* The deformed coordinates can be related to the material coordinates and the displacement vector:

$$\boldsymbol{x}(\boldsymbol{X}, t) = \boldsymbol{X} + \boldsymbol{u}(\boldsymbol{X}, t)$$

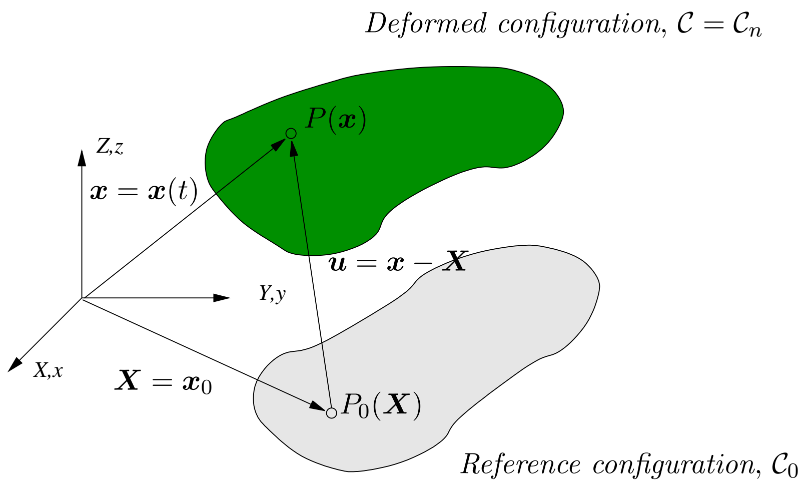



Figure 11.1: The figure show a geometric nonlinear system using a Lagrangian formulation: coordinate system, reference- and deformed- configuration, and the displacements.

The differential quantities are related through the deformation gradient:

$$d\boldsymbol{x} = \frac{\partial \boldsymbol{x}}{\partial \boldsymbol{X}} d\boldsymbol{X} = \frac{\partial(\boldsymbol{X} + \boldsymbol{u})}{\partial \boldsymbol{X}} d\boldsymbol{X} = \Big(\boldsymbol{I} + \frac{\partial \boldsymbol{u}}{\partial \boldsymbol{X}}\Big) d\boldsymbol{X}$$

### 11.2.2. Strains

In nonlinear models a number of definition of strains are used. In a total Lagrangian formulation the strains are the *Green-Lagrange strains*, while other strains are combined with the alternative models. Here are some strains used in modeling nonlinear solid mechanics:

- Biot or rotated engineering strains,
- Green-Lagrange strains,
- Logarithmic strains,
- Almansi strains,
- etc.

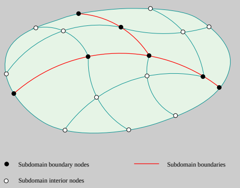
Common properties for all strains are:

- Zero strains when

$$||d\boldsymbol{x}|| = ||d\boldsymbol{X}||$$

  i.e. for for rigid body displacements.

Department of Mathematics University of Oslo




Mek 4560
Torgeir Rusten

- Identical for small strains.
- Monotonically increasing.

The Green-Lagrange strains $\boldsymbol{E}$, can be expressed using the deformation gradient:

$$\boldsymbol{E} = \frac{1}{2}(\boldsymbol{F}^T\boldsymbol{F} - \boldsymbol{I}) = \frac{1}{2}\left(\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{X}} + \left(\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{X}}\right)^T + \left(\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{X}}\right)^T\left(\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{X}}\right)\right)$$

Note that the strain tensor is a nonlinear function of the displacements, $\boldsymbol{u}$. Furthermore, $\boldsymbol{F}^T\boldsymbol{F} = \boldsymbol{I}$ for rigid body deformations, thus the strain measure is zero for rigid body displacements.

The component for of the Green-Lagrange strains are

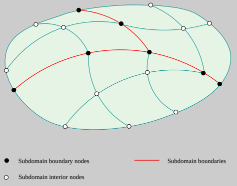
$$E_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial X_j} + \frac{\partial u_j}{\partial X_i} + \frac{\partial u_k}{\partial X_i}\frac{\partial u_k}{\partial X_j}\right)$$

**Remark 11.1** Note that for sufficiently small deformations, we obtain the linearized strain tensor.

### 11.2.3. Stress

We mentioned above that a number of different definitions of strains is in use. The energy in the potential energy function or the work in the virtual work is unique physical quantities,

Department of Mathematics
University of Oslo






Contents
Page 18 of 38
Go Back
Close
Quit

thus the definition of the stress must be consistent with the definition of the strain. The stress definition corresponding to the Green-Lagrange strains are the 2. Piola-Kirchoff stresses. Other definition of strains is combined with other definitions of the stresses:

- Cauchy stresses (true stresses),
- Kirchhoff stresses (nominal stresses),
- First Piola-Kirchhoff stresses,
- etc.

The relation between the different stresses can be found from the virtual work:

$$\begin{aligned}\int_{V(t)} \boldsymbol{\sigma} : \delta\boldsymbol{\varepsilon}\, dV &= \int_{V(t)} \boldsymbol{\sigma} : \nabla_{\boldsymbol{x}}\delta\boldsymbol{u}\, dV = \int_{V(0)} J\boldsymbol{\sigma} : \nabla_{\boldsymbol{x}}\delta\boldsymbol{u}\, dV = \int_{V(0)} \boldsymbol{\tau} : \nabla_{\boldsymbol{x}}\delta\boldsymbol{u}\, dV \\ &= \int_{V(0)} J\boldsymbol{\sigma}\cdot\boldsymbol{F}^{-T}\cdot\boldsymbol{F}^{T} : \nabla_{\boldsymbol{x}}\delta\boldsymbol{u}\, dV = \int_{V(0)} \boldsymbol{P} : \delta\boldsymbol{F}^{T}\, dV \\ &= \int_{V(0)} \boldsymbol{F}\cdot\boldsymbol{F}^{-1}\boldsymbol{P} : \delta\boldsymbol{F}^{T}\, dV = \int_{V(0)} \boldsymbol{S} : \delta\boldsymbol{E}\, dV\end{aligned}$$

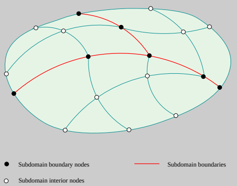
Above we used the Jacobian $J$, it relates the volume of the deformed and initial body:

$$J = \det\boldsymbol{F}, \qquad dV = J\, dV_0$$

This give the following relations between the stresses:

$$J\boldsymbol{\sigma} = \boldsymbol{\tau} = \boldsymbol{P}\boldsymbol{F}^{T} = \boldsymbol{F}\boldsymbol{S}\boldsymbol{F}^{T}$$





where

| | |
|---|---|
| $\boldsymbol{\sigma}$ | *Cauchy* stresses or “true” stresses. |
| $\boldsymbol{\tau}$ | *Kirchhoff* stresses, or nominal stresses. |
| $\boldsymbol{P}$ | First Piola-Kirchhoff stresses. |
| $\boldsymbol{S}$ | Second Piola-Kirchhoff stresses. |

## 11.2.4. Equilibrium equations and linearization

Using the 2. Piola-Kirchhoff stresses and the Green-Lagrange stresses the equilibrium equations becomes:

$$\delta\Pi(\boldsymbol{u};\delta\boldsymbol{u}) = \int_{V(0)} \boldsymbol{S} : \delta\boldsymbol{E}\, dV - \int_{V(0)} \bar{\boldsymbol{F}} \cdot \delta\boldsymbol{u}\, dV - \int_{S(0)} \bar{\boldsymbol{\Phi}} \cdot \delta\boldsymbol{u}\, dS = 0$$

It is clear that the virtual internal work

$$\delta W^i = \int_{V(0)} \boldsymbol{S} : \delta\boldsymbol{E}\, dV$$

is a nonlinear function of $\boldsymbol{u}$. A Finite Element formulation result in a nonlinear system of algebraic equations in the nodal displacements, $\boldsymbol{D}$. The global equilibrium equation takes the form:

$$\boldsymbol{K}(\boldsymbol{D})\boldsymbol{D} = \boldsymbol{R}$$

The expressions can be linearized and used in linearized buckling or Newtons method for the solution of nonlinear algebraic equation. The linearized expressions are:

$$\Delta\delta\Pi(\boldsymbol{u};\delta\boldsymbol{u};\Delta\boldsymbol{u}) = \int_{V(0)} \Delta\boldsymbol{S} : \delta\boldsymbol{E}\,dV + \int_{V(0)} \boldsymbol{S} : \Delta\delta\boldsymbol{E}\,dV - \int_{V(0)} \Delta\bar{\boldsymbol{F}} \cdot \delta\boldsymbol{u}\,dV - \int_{S(0)} \Delta\bar{\boldsymbol{\Phi}} \cdot \delta\boldsymbol{u}\,dS = 0$$

The linearized internal virtual work

$$\Delta\delta W^i = \int_{V(0)} \Delta\boldsymbol{S} : \delta\boldsymbol{E}\,dV + \int_{V(0)} \boldsymbol{S} : \Delta\delta\boldsymbol{E}\,dV$$

is a nonlinear function of $\boldsymbol{u}$. In a element formulation this result in a nonlinear expression for the nodal displacements, $\boldsymbol{D}$. The *incremental* equilibrium equations can be written:

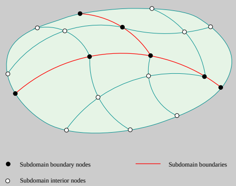
$$\boldsymbol{K}_T(\boldsymbol{D})\Delta\boldsymbol{D} = \Delta\boldsymbol{R} \qquad \text{where} \qquad \boldsymbol{K}_T = \boldsymbol{K}_m + \boldsymbol{K}_g$$

$\boldsymbol{K}_m$ is the material stiffness matrix and $\boldsymbol{K}_g$ the geometric stiffness matrix.

**Remark 11.2** The geometric stiffness matrix depend on the displacements, thus in order to use i for linearized buckling the displacements must be known.

**Remark 11.3** As indicated above, the equilibrium equations can be expressed in different coordinate systems.

Department of Mathematics University of Oslo




Mek 4560
Torgeir Rusten

## 11.3. Stress strain relation

The stress strain relation defines the properties of the material and relate the *kinematic* quantities(strains) and the *forces* (stresses). They are found using experiments. Note that the relations must satisfy some constraints resulting from thermodynamical arguments.

Note that since different stress and strain measures are used depending on the coordinate system, it is important to use the correct set of material parameters.

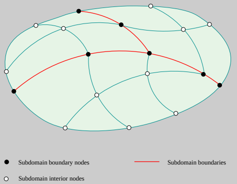
The material laws can also be transformed between coordinate systems. reference system.

## 11.4. Bar analysis

**Introduction:** In this section we discuss a bar analysis. The nonlinear equations are expressed using the *total Lagrangian formulation*, the *Green-Lagrange strains* and the corresponding *second Piola-Kirchhoff stresses*.

The bar is one-dimensional, and we work in a one dimensional space below. For analysis of three dimensional constructions made of a set of bars the element matrices can be transformed to global element matrices. This is briefly considered at the end of the sections.








Contents
Page 22 of 38
Go Back
Close
Quit

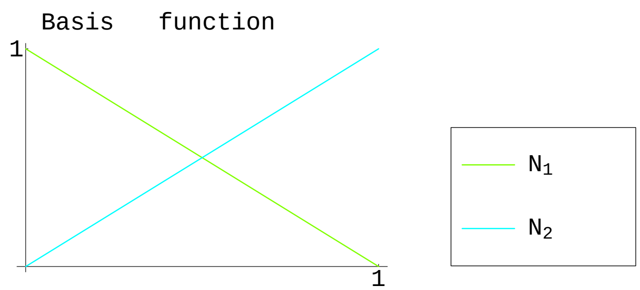
**Interpolated displacements:** The polynomials are based on Lagrange interpolation, where the nodal points are the endpoints of the bar$(0, \ell)$. For linear interpolation:

$$u = \boldsymbol{N}\boldsymbol{d}$$

where

$$\boldsymbol{N} = \begin{Bmatrix} -\frac{X-\ell}{\ell} & \frac{X}{\ell} \end{Bmatrix} \qquad \text{and} \qquad \boldsymbol{d} = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

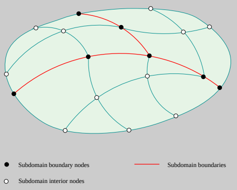
Here physical coordinates are used. The basis functions are shown for $(\ell = 1)$:




**The deformation gradient:** The deformation gradient is:

$$\boldsymbol{F}_{xX} = F_{xX} = F = 1 + \frac{du}{dX} = 1 - \frac{u_1}{\ell} + \frac{u_2}{\ell}$$





**Green strains:** The Green-Lagrange strains can be computed when the deformation gradient is given:

$$E_{XX} = \frac{1}{2}\left(F^2 - 1\right) = \frac{-\left(u_1 - u_2\right)\,\left(2\,\ell - u_1 + u_2\right)}{2\,\ell^2}$$

The virtual strains are given by:

$$\delta E_{XX} = \left(\frac{u_1 - u_2}{2\,\ell^2} - \frac{2\,\ell - u_1 + u_2}{2\,\ell^2}\right)\,\delta u_1 + \left(\frac{-\left(u_1 - u_2\right)}{2\,\ell^2} + \frac{2\,\ell - u_1 + u_2}{2\,\ell^2}\right)\,\delta u_2$$

And the corresponding strain increments:

$$\Delta E_{XX} = \left(\frac{u_1 - u_2}{2\,\ell^2} - \frac{2\,\ell - u_1 + u_2}{2\,\ell^2}\right)\,\Delta u_1 + \left(\frac{-\left(u_1 - u_2\right)}{2\,\ell^2} + \frac{2\,\ell - u_1 + u_2}{2\,\ell^2}\right)\,\Delta u_2$$

The second order expressions are:

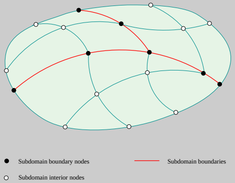
$$\Delta\delta E_{XX} = \left(\frac{\delta u_1}{\ell^2} - \frac{\delta u_2}{\ell^2}\right)\,\Delta u_1 + \left(-\left(\frac{\delta u_1}{\ell^2}\right) + \frac{\delta u_2}{\ell^2}\right)\,\Delta u_2$$

**The stiffness matrix:** The stiffness matrix is found from the general expressions, the material stiffness matrix is given by (the axial stiffness is constant EA):

$$\int_\ell \Delta S_{XX}\delta E_{XX}\,dX = EA\int_\ell \delta E_{XX}\Delta E_{XX}\,dX = \delta\boldsymbol{d}^T\boldsymbol{k}_m\Delta\boldsymbol{d} \qquad \text{where}$$

$$\boldsymbol{k}_m = \frac{EA}{\ell^3}\begin{bmatrix} \left(\ell - u_1 + u_2\right)^2 & -\left(\left(\ell - u_1 + u_2\right)^2\right) \\ -\left(\left(\ell - u_1 + u_2\right)^2\right) & \left(\ell - u_1 + u_2\right)^2 \end{bmatrix}$$





The geometric stiffness matrix is given by:

$$\int_\ell S_{XX}\Delta\delta E_{XX}\,dX = EA\int_\ell E_{XX}\Delta\delta E_{XX}\,dX = \delta\boldsymbol{d}^T\boldsymbol{k}_g\Delta\boldsymbol{d} \qquad \text{where}$$

$$\boldsymbol{k}_g = EA\begin{bmatrix} \frac{-((u_1-u_2)\,(2\,\ell-u_1+u_2))}{2\,\ell^3} & \frac{(u_1-u_2)\,(2\,\ell-u_1+u_2)}{2\,\ell^3} \\ \frac{(u_1-u_2)\,(2\,\ell-u_1+u_2)}{2\,\ell^3} & \frac{-((u_1-u_2)\,(2\,\ell-u_1+u_2))}{2\,\ell^3} \end{bmatrix}$$

To obtain the total stiffness matrix add the material and geometric stiffness matrices:

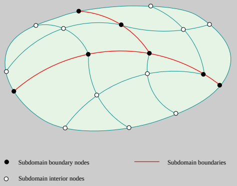
$$\boldsymbol{k}_T = \boldsymbol{k}_m + \boldsymbol{k}_g = EA\begin{bmatrix} \frac{2\,\ell^2+3\,(u_1-u_2)\,(-2\,\ell+u_1-u_2)}{2\,\ell^3} & \frac{-2\,\ell^2+3\,(u_1-u_2)\,(2\,\ell-u_1+u_2)}{2\,\ell^3} \\ \frac{-2\,\ell^2+3\,(u_1-u_2)\,(2\,\ell-u_1+u_2)}{2\,\ell^3} & \frac{2\,\ell^2+3\,(u_1-u_2)\,(-2\,\ell+u_1-u_2)}{2\,\ell^3} \end{bmatrix}$$

**Internal load vector:** The internal load vector can be established from the internal virtual work:

$$\int_\ell S_{xx}\delta E_{xx}\,dX = EA\int_\ell E_{xx}\delta E_{xx}\,dX = \delta\boldsymbol{d}^T\boldsymbol{r}^i \qquad \text{where} \qquad \boldsymbol{r}^i = \begin{Bmatrix} \frac{(u_1-u_2)\,(\ell-u_1+u_2)\,(2\,\ell-u_1+u_2)}{2\,\ell^3} \\ \frac{-((u_1-u_2)\,(\ell-u_1+u_2)\,(2\,\ell-u_1+u_2))}{2\,\ell^3} \end{Bmatrix}$$

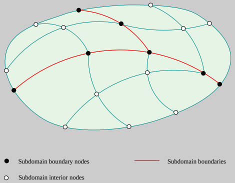
**Load-displacement relation:** The load-displacement relation for a bar loaded by an axial forces:

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

Department of Mathematics
University of Oslo





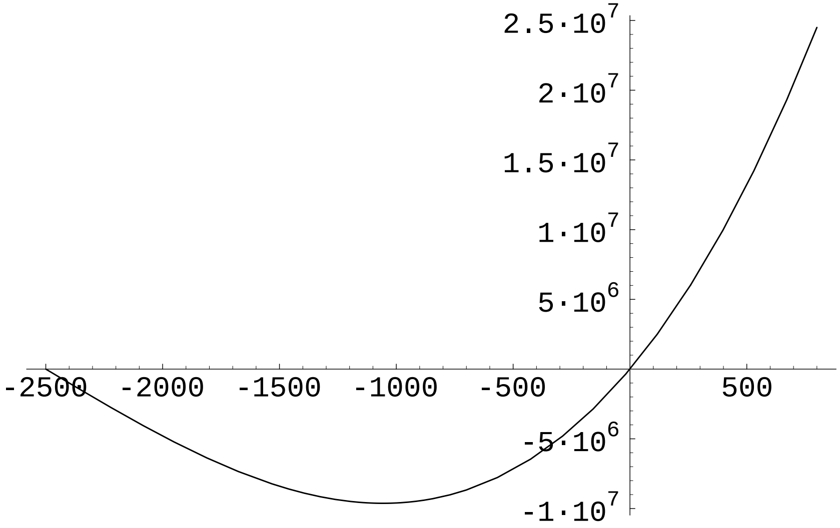

Contents

Page 26 of 38

Go Back

Close

Quit




**Transformations to a global system:** If the element is part of a three dimensional model the local element matrices can be transformed to a global system. The transformation matrix is given by the expression for the tangent vector. For a straight bar this is constant along the bar axis and is found from:

$$\hat{\boldsymbol{t}} = \frac{\partial \boldsymbol{x}}{\partial \xi} \qquad \boldsymbol{t} = \frac{\hat{\boldsymbol{t}}}{\sqrt{\hat{\boldsymbol{t}} \cdot \hat{\boldsymbol{t}}}}$$

Consequently, the transformation from the global to local coordinate system is

$$\boldsymbol{T}_0 = \boldsymbol{t}^T$$

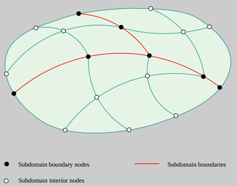
This is a $1 \times 3$ matrix. The local transformation matrix for the element stiffness matrix and the load vector is:

$$\boldsymbol{T} = \begin{bmatrix} \boldsymbol{T}_0 & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{T}_0 \end{bmatrix} = \begin{bmatrix} \boldsymbol{t}^T & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{t}^T \end{bmatrix} \qquad \text{which is } 2 \times 6$$

The global quantities are:

$$\begin{aligned} \boldsymbol{k}_m^g &= \boldsymbol{T}^T \boldsymbol{k}_m \boldsymbol{T} && \text{which is } 6 \times 6 \\ \boldsymbol{k}_g^g &= \boldsymbol{T}^T \boldsymbol{k}_g \boldsymbol{T} && \text{which is } 6 \times 6 \\ \boldsymbol{r}^{ig} &= \boldsymbol{T}^T \boldsymbol{r}^i && \text{which is } 3 \times 1 \end{aligned}$$





## 11.5. Bjelkebøyning (BEAM3)

**Problem:** Den samme konstruksjonen som i Kapittel 2 *BEAM3* skal analyseres med ANSYS. Store forskyvninger/tøyninger skal tas hensyn til og lasten påføres inkrementvis. Er den vertikale forskyvning av punkt $A$ den samme som før?

**Løsning:**

```
/BATCH,LIST
/FILNAM, ex451
/TITLE,  Ikke-lineær statisk analyse av en bjelke

/PREP7                  ! Preprossessoren
ET,1,BEAM3              ! BEAM3 elementer
R,1,500,1.936e9,6.5e3   ! Høyde, I-verdi og areal til tverrsnitt
MP,EX,1,2e5             ! E-modulen

!Geometri ("solid modelling")
K,1,0                ! Punktene A er origo
K,2,4e3              !          B
K,3,12e3             !          C
K,4,20e3             !      og  D

L,1,2                ! Linje AB
L,2,3                !       BC
L,3,4                !   og  CD

!Inndeling i elementer
LSEL,S,line,,2,3       ! Velger linjene BC og CD
LESIZE,all,,,8         ! som inndeles i 8 elementer hver
LSEL,inve              ! Velger invers av det valgte setet (dvs. linje AB)
```

Department of Mathematics
University of Oslo

Mek 4560
Torgeir Rusten

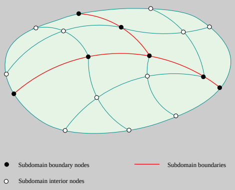
```
LESIZE,all,,,4        ! som deles inn i 4 elementer
LSEL,all              ! Velger all linjer igjen

REAL,1 $ TYPE,1 $ MAT,1 ! Egentlig ikke noedvendig (de er default)
LMESH,1,3               ! Inndeling av alle linjer
FINISH                  ! Ut av Preprossessoren

/SOLU                   ! Løsningsprossessoren
ANTYPE,STAT             ! Statisk analyse
NLGEOM,ON               ! Ikke-lineær geometrisk analyse
D,NODE(4E3,0,0),UX,,,,,UY   ! Opplagring ved B
D,NODE(20E3,0,0),UY
ESEL,S,ELEM,,1,4            ! Velger elementene mellom A og B
SFBEAM,ALL,1,PRES,150       ! Jevnt fordelt lasten
!(PSF,PRES,NORM,2 og eplo for å se på lasten)
ESEL, ALL
F,NODE(12E3,0,0),MZ,-12e9   ! Momentet paa C
!TIME,150                   ! Skaler "time" til fordelte lasten
AUTOTS,ON                   ! Automatisk inkrementering av lasten
PRED,ON
LNSRCH,ON
NSUBST,10,100,5             ! Lastinkrement
KBC,0                       ! Stigende/minkende last
NCNV,2
NEQIT,25                    ! Begrensning på antall iterasjoner
OUTRES,ALL,ALL              ! Resultater for alle lastinkrementer
SOLVE                       ! Løsningsprosedyren
FINISH                      ! Ut av Løsningsprossessoren

/POST1                ! Postprossessoren
SET,LAST              ! Last inn resultat for siste lastinkrement
PLDISP,1              ! Deformert konstruksjon
PRESOL,F              !  .. gir skjærkreftene
PRESOL,M              !  .. gir momentene
PRNSOL,U              ! Print av nodeforskyvningene
FINISH
```

**Kommentarer:** Merk at kommandoen `NLGEOM` alltid må brukes for ikke-lineære analyser. Automatisk inkrementering av lasten anvendes med `AUTOTS` og `NSUBST` kommandoene deklarerer minimum og maksimum verdier av inkrementene.

**Svar på spørsmålene:** $M$- og $V$-diagrammene tilsvarer de i Kapittel 2 *BEAM3*. De vertikale forskyvningene av punktene $A$ og $C$ er henholdsvis $-161.07mm$ og $49.533mm$ for $q = 150kN/m$ og $-860.85mm$ og $494.86mm$ for $q = 1500kN/m$.

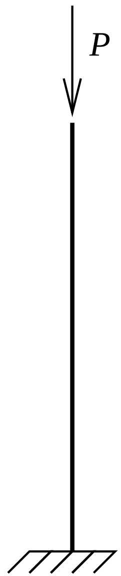
Sammenligning med resultatene fra Kapittel 2 *BEAM3* viser at for lavere laster (som forårsaker relativt små forskyvninger) er vertikalforskyvningene av punktene $A$ og $C$ nesten lik de fra den lineære analysen. Når lasten er stor og det oppstår relativ store forskyvninger, så avviker resultatene fra de to analysene.

Department of Mathematics
University of Oslo



## 11.6. Stor forskyvning av en aksialbelastet stav

*Elastica [Timoshenko and Gere, 1963]) (BEAM3)*

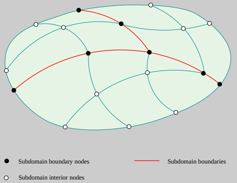



$P_{kr} = \frac{\pi^2 EI}{4\ell^2} = 38.553$

$E = 3 \times 10^7 \; A = 0.25$

$I = 5.208 \times 10^{-3} \; l = 100$

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

**Problem:** Figuren viser en slank stav/søyle som belastes med med lasten $P > P_{kr}$. Beregn forskyvningen av staven ved

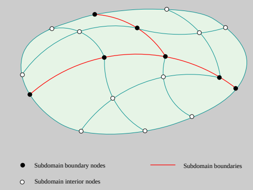
$$P/P_{kr} = \{1.015,\ 1.063,\ 1.152,\ 1.293,\ 1.518,\ 1.884\}$$

Plot den deformerte staven for lastene ovenfor (i kun én figur).

**Løsning:**

```
/BATCH,LIST
/FILNAM, ex452
/TITLE, Stor forskyvning av en aksialbelastet stav (Elastica)

/PREP7
ET,1,BEAM3,,,,,,,1
R,1,.25,52083E-7,.5
MP,EX,1,3E7
N,1
N,11,,100
FILL
E,1,2
EGEN,10,1,1
FINISH

/SOLUTION
ANTYPE,STATIC
NLGEOM,ON
NEQIT,150
OUTPR,BASIC,LAST      ! Antall tillatte iterasjoner
D,1,ALL
PCR=-38.553           ! Kritisk last
F,11,FY,PCR*1.015     ! Vertikal last
```



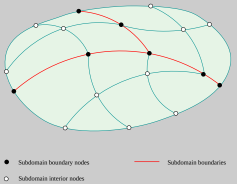


```
F,11,FX,.5            ! Pertubasjonslast
SOLVE
FDEL,11,FX            !fjern pertubasjonslast
F,11,FY,PCR*1.063
SOLVE
F,11,FY,PCR*1.152
SOLVE
F,11,FY,PCR*1.293
SOLVE
F,11,FY,PCR*1.518
SOLVE
F,11,FY,PCR*1.884
SOLVE
FINISH

/POST1
/SHOW, ex452pl
/USER
/FOCUS,,50,50
/DIST,,55
/DSCALE,,1
/PLOPTS,FRAME,OFF
/PLOPTS,INFO,OFF
/PLOPTS,TITLE,OFF

SET,1,0  $PLDISP,1  ! Vise udeformert geometri
/NOERASE
*DO,I,2,6,1
   SET,I,0
   PLDISP
*ENDDO
FINISH
```

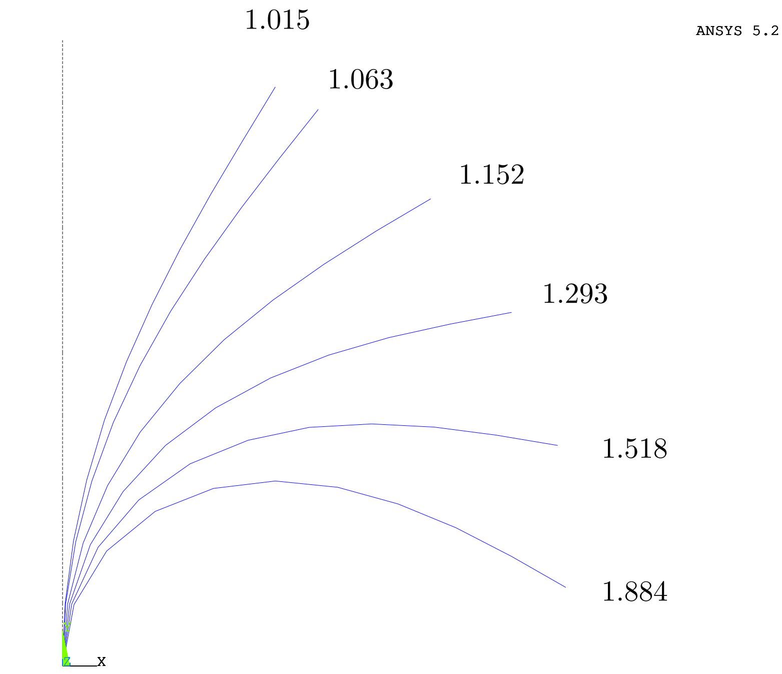
**Kommentarer:** Her er det brukt en annen måte å inkrementere belastningen. En pertubasjonslast må til for at forskyvning ut av bjelkens akse kan være mulig.

Figuren viser plott av den deformerte konstruksjonen for de forskjellige lasttilfellene.

Department of Mathematics University of Oslo




Mek 4560
Torgeir Rusten




Deformert geometri av en aksialbelastet stav for lasttilfellene $P_{kr} \times k$ ($k$ er tallene vist i figuren)

## Øving 11.1

Et stav element med lengde $\ell_0$, som opprinnelig ligger langs $X \equiv x$ aksen, roteres til å ligge langs $Y \equiv y$ aksen. Node én ligger i origo $X = Y = 0$ også etter deformasjonen. Deformasjonen er generelt gitt ved

$$x(X,Y,Z) = -Y, \qquad y(X,Y,Z) = X \qquad \text{og} \qquad z(X,Y,Z) = Z$$

a) Beregn forskyvningsfeltet $\boldsymbol{u} = (u, v, z)$.

b) Beregn deformsjonsgradienten, $\boldsymbol{F}$. Benytt uttrykket

$$\boldsymbol{F} = \frac{\partial \boldsymbol{x}}{\partial \boldsymbol{X}} = \begin{bmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} & \frac{\partial x}{\partial Z} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} & \frac{\partial y}{\partial Z} \\ \frac{\partial z}{\partial X} & \frac{\partial z}{\partial Y} & \frac{\partial z}{\partial Z} \end{bmatrix} = \begin{bmatrix} F_{xX} & F_{xY} & F_{xZ} \\ F_{yX} & F_{yY} & F_{yZ} \\ F_{zX} & F_{zY} & F_{zZ} \end{bmatrix}$$

c) Beregn aksial tøyningen, $E_{xx}$ (Green-Lagrange tøyningen). Dette er stavens eneste tøyningskomponent. Benytt formelen

$$E_{xx} = \frac{1}{2}(F_{xX}^2 + F_{yX}^2 + F_{zX}^2 - 1)$$

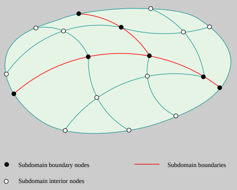
Gir ren rotasjon tøyning?

d) Benytt uttrykket for ingeniørtøyning for dette problemet

$$\varepsilon_{xx} = \frac{\partial u}{\partial X}$$

Gir dette en fornuftig tøyning?

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

## Øving 11.2

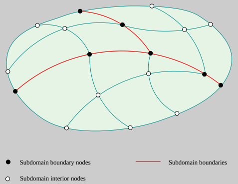
Anta at den uroterte staven i Oppgave 11.1 er utsatt for strekk, $S_{xx} = S_{xx0}$.

- Hva er spenningen $S_{XX}$ i den roterte staven?
- Benytt sammenhengen

$$J\boldsymbol{\sigma} = \boldsymbol{F}\boldsymbol{S}\boldsymbol{F}^T$$

til å finne de sanne, Cauchy, spenningene i den roterte staven.

Department of Mathematics
University of Oslo




Mek 4560
Torgeir Rusten

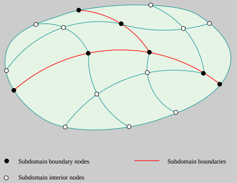
# A. References


[Cook et al., 2002] Cook, R. D., Malkus, D. S., Plesha, M. E., and Witt, R. J. (2002). *Concepts and Applications of Finite Element Analysis*. Number ISBN: 0-471-35605-0. John Wiley & Sons, Inc., 4th edition.

[Timoshenko and Gere, 1963] Timoshenko, S. P. and Gere, J. M. (1963). *Theory of Elastic Stability*. McGraw-Hill, N.Y., 2 edition.



Department of Mathematics University of Oslo



Mek 4560
Torgeir Rusten